

IT'14
ŽABLJAK

XIX

naučno - stručni skup

INFORMACIONE TEHNOLOGIJE

SADAŠNJOST I BUDUĆNOST

Urednik
Božo Krstajić

IT'14

INFORMACIONE TEHNOLOGIJE

- SADAŠNJOST I BUDUĆNOST -

Urednik
Božo Krstajić

Zbornik radova sa XIX naučno - stručnog skupa
INFORMACIONE TEHNOLOGIJE - sadašnjost i budućnost
održanog na Žabljaku od 24. do 28. februara 2014. godine

Zbornik radova
INFORMACIONE TEHNOLOGIJE - sadašnjost i budućnost 2014

Glavni urednik
Prof.dr Božo Krstajić

Izdavač
Univerzitet Crne Gore
Elektrotehnički fakultet
Džordža Vašingtona bb., Podgorica
www.etf.ucg.ac.me

Tehnička obrada
Aleksandra Radulović
Centar Informacionog Sistema
Univerziteta Crne Gore

Tiraž
150

Podgorica 2014.

Sva prava zadržava izdavač i autori

Organizator

Elektrotehnički fakultet Univerziteta Crne Gore

Skup su podržali:

- * Ministarstvo za informaciono društvo i telekomunikacije
- * Pošta Crne Gore
- * Agencija za elektronske komunikacije i poštansku djelatnost
- * Čikom

Programski odbor

Dr Novak Jauković, Elektrotehnički fakultet, Podgorica
Akademik Dr Ljubiša Stanković, CANU
Dr Zdravko Uskoković, Elektrotehnički fakultet, Podgorica
Dr Vujica Lazović, Ekonomski fakultet, Podgorica
Dr Branko Kovačević, Elektrotehnički fakultet, Beograd
Dr Milorad Božić, Elektrotehnički fakultet, Banja Luka
Dr Miroslav Bojović, Elektrotehnički fakultet, Beograd
Dr Zoran Jovanović, Elektrotehnički fakultet, Beograd
Dr Božidar Krstajić, Elektrotehnički fakultet, I. Sarajevo
Dr Milica Pejanović-Đurišić, Elektrotehnički fakultet, Podgorica
Dr Dejan Popović, Elektrotehnički fakultet, Beograd
Dr Božo Krstajić, Elektrotehnički fakultet, Podgorica
Dr Milovan Radulović, Elektrotehnički fakultet, Podgorica
Dr Budimir Lutovac, Elektrotehnički fakultet, Podgorica
Dr Igor Radusinović, Elektrotehnički fakultet, Podgorica
Dr Igor Đurović, Elektrotehnički fakultet, Podgorica
Dr Miloš Daković, Elektrotehnički fakultet, Podgorica
Dr Milutin Radonjić, Elektrotehnički fakultet, Podgorica
Dr Ramo Šendelj, Fakultet za Informacione Tehnologije, Podgorica
Dr Stevan Šćepanović, Prirodno-matematički fakultet, Podgorica
Dr Sašo Gelev, Elektrotehnički fakultet, Radoviš

Organizacioni odbor

Dr Božo Krstajić, Elektrotehnički fakultet, Podgorica
Dr Milovan Radulović, Elektrotehnički fakultet, Podgorica
Dr Ana Jovanović, Elektrotehnički fakultet, Podgorica
Dr Saša Mujović, Elektrotehnički fakultet, Podgorica
MSc Žarko Zečević, Elektrotehnički fakultet, Podgorica
Vladan Tabaš, dipl.ing., Čikom

Sekretarijat

Aleksandra Radulović, CIS Univerzitet Crne Gore

SADRŽAJ

Vesna Rubežić, Ana Jovanović HAOS U LASERSKOM SISTEMU SA ERBIJUM-DOPIRANIM VLAKNOM.....	1
Žarko Zečević, Igor Đurović, Božo Krstajić DISTRIBUIRANI SET-MEMBERSHIP NLMS ALGORITAM	5
Žarko Zečević, Božo Krstajić BRZI COUPLED LMS ALGORITAM.....	9
Luka Lazović, Ana Jovanović, Vesna Rubežić UPOREDNA ANALIZA PERFORMANSI CAPON I CAPON-LIKE ALGORITAMA U SISTEMIMA PAMETNIH ANTENA	13
Mladen Rašović, Jadranka Radović, Saša Mujović MODELOVANJE SVJETILJKI PRI ANALIZAMA KVALITETA ELEKTRIČNE ENERGIJE U MREŽAMA JAVNE RASVJETE	17
Novica Daković, Milovan Radulović FLATNESS UPRAVLJANJE AUTONOMNO VOĐENIM VOZILOM	21
Balša Femić, Stevan Šćepanović PRIVATNI OBLAK OTVORENOG KODA.....	25
Luka Filipović, Božo Krstajić PREDLOG POBOLJŠANJA MASTER-SLAVE ALGORITMA ZA RASPODJELU OPTEREĆENJA U MPI PARALELNIM APLIKACIJAMA.....	29
Sidita Duli HIBRIDNA MPI/PTHREADS PARALELIZACIJA ZA ESTIMACIJU PARAMETARA WEIBULL DISTRIBUCIJE	33
Uglješa Urošević, Zoran Veljović POBOLJŠANJE ENERGETSKE EFIKASNOSTI OFDM-CDMA SISTEMA	36
Srdjan Jovanovski, Veselin N. Ivanović PIPELINE-OVANI SISTEM ZA ESTIMACIJU VISOKO NESTACIONARNIH FM SIGNALA	40
Borko Drašković INTEGRACIJA SaaS SERVISA U CLOUD TELEKOMA SRBIJA	44
Nemanja Filipović, Radovan Stojanović MONITORING I ANALIZA VITALNIH FIZIOLOŠKIH PARAMETARA PRIMJENOM PDA UREĐAJA	48
Roman Golubovski KONCEPT ZA EKSPERTNI SISTEM ZA AUTOMATIZOVANU EKG DIJAGNOSTIKU.....	52

Nataša Savić, Zoran Milivojević, Darko Brodić ANALIZA EFIKASNOSTI KVADRATNIH KONVOLUCIONI H JEZGARA KOD PROCENE FREKVENCIJE SIGNALA	56
Igor Ivanović, Srđan Kadić UPOTREBA OpenFlow STANDARDA ZA BALANSIRANJE NFS SERVERA SA SISTEMOM POVRATNE SPREGE.....	60
Miladin Tomić, Milutin Radonjić, Neđeljko Lekić, Igor Radusinović VIRTUELIZACIJA MREŽE KORIŠĆENJEM ALATA FLOWVISOR	64
Slavica Tomović, Milutin Radonjić, Igor Radusinović IMPLEMENTACIJA RIP I OSPF PROTOKOLA NA QUAGGA SOFTVERSKOJ PLATFORMI.....	68
Rabina Šabotić, Milutin Radonjić, Igor Radusinović DISTRIBUCIJA UNIVERZALNOG KOORDINISANOG VREMENA.....	72
Neđeljko Lekić, Almir Gadžović, Igor Radusinović V2X SISTEMI KOOPERATIVNE MOBILNOSTI.....	76
Jelena Šuh, Vladimir Ćulum PROTOKOLI ZA REDUDANSU U IP MREŽAMA	80
Pero Bogojević, Jasna Mirković MOBILNA APLIKACIJA CRNOGORSKOG TELEKOMA	84
Mirko Kosanović, Miloš Kosanović INTEGRACIJA BEŽIČNIH SENZORSKIH MREŽA U CLOUD COMPUTING-u	88
Aleksandar Trifunović, Svetlana Čičević, Andreja Samčović, Milkica Nešić PRIMENA TABLET TEHNOLOGIJE U SAVLAĐIVANJU REČI ENGLESKOG JEZIKA	92
Aleksandar Ristić, Dalibor Damjanović OPRAVDANOST INICIJATIVE ZA IZMJENU NASTAVNOG PLANA I PROGRAMA INFORMATIKE U SREDNJEM STRUČNOM OBRAZOVANJU EKONOMSKE I TRGOVINSKE STRUKE, ZANIMANJE TRGOVAC`	96
Risto Bojović MODEL STRATEŠKOG RAZMIŠLJANJA U IT OKRUŽENJU	100
Vuko Perišić STRATEŠKO PLANIRANJE ICT-A U SKUPŠTINI CRNE GORE.....	104
Željko Pekić, Stevan Kordić, Draško Kovač, Tatijana Dlabač, Nađa Pekić ANALIZA ONLINE KOMUNIKACIJE I INTERAKCIJE KROZ E-LEARNING.....	108
Dejan Abazović, Budimir Lutovac ITIL - IMPLEMENTACIJA INCIDENT MANAGEMENT PROCESA U SERVICE DESK-U SA PREDLOGOM ZA NJEGOVO UNAPREĐENJE	112

Jelena Končar, Sonja Leković IMPLEMENTACIJA INTERAKTIVNE ELEKTRONSKE MALOPRODAJE U REPUBLICI SRBIJI	116
Obradović Milovan EVOLUCIJA SISTEMA ZA PODRŠKU ODLUČIVANJU I NJIHOVE PRIMENE U ZDRAVSTVU	120
Ilija Apostolov, Risto Hristov, Sašo Gelev IZBOR OPTIMALNE TEHNIKE ZA ENKRIPCiju I DEKRIPCiju PODATAKA.....	124
Tamara Pejaković, Miloš Orović, Andjela Draganić, Irena Orović INFORMACIONI SISTEM ZA IZVJEŠTAVANJE O PRODUKTIVNOSTI AGENATA OSIGURAVAJUĆEG DRUŠTVA.....	128
Predrag Raković, Vasilije Stijepović ANALIZA POTREBA ZA PRIKUPLJANJE PODATAKA, EVALUACIJU I PRAĆENJE SPORTSKIH POVREDA U CRNOGORSKIM SPORTSKIM KLUBOVIMA I SAVEZIMA.....	132
Bogdan Mirković ISTRAŽIVANJE NEFUNKCIONALNIH ZAHTJEVA INFORMACIONIH SISTEMA	135
Bogdan Mirković KRITERIJUMI ZA MJERENJE USPJEŠNOSTI INFORMACIONIH SISTEMA.....	139
Sanja Bauk, Tatijana Dlabač, Radoje Džankić O AMOS SOFTVERU NAMIJENJENOM ELEKTRONSKOM UPRAVLJANJU RESURSIMA NA BRODU	143
Predrag Raković IMPLEMENTACIJA MPI ZA UBRZANJE ESTIMACIJE PARAMETARA 2DCPPPS PRIMJENOM 2DCPF-A	147
Milovan Radulović, Vesna Rubežić, Martin Čalasan HAOTIČNI OPTIMIZACIONI METOD SINTEZE PID REGULATORA U AVR SISTEMU	150
Tamara Bojičić, Vesna Popović-Bugarin UTICAJ KRITERIJUMA MINIMIZACIJE NA UPRAVLJANJE POTROŠNJOM ELEKTRIČNE ENERGIJE	154
Mirjana Božović, Saša Mujović PRIMJENA SAVREMENOG MJERNO-AKVIZICIONOG SISTEMA ZA MONITORING KVALITETA ELEKTRIČNE ENERGIJE NA PRIMJERU ŽELJEZARE NIKŠIĆ	158
Ana Grbović, Bojan Đordan IMPLEMENTACIJA I VIZUELIZACIJA GRUPNE REGULACIJE U HE PERUĆICA	162
Roman Golubovski JEFTINO PIC BAZIRANO REŠENJE ZA AUTO-TRAKING FOTONAPONSKIH PANELA.....	166

Saša Stojanović, Dragan Tošić, Zoran Milivojević SUN TRACKER SISTEM BAZIRAN NA MIKROKONTROLERU	170
Dimitrija Angelkov, Cveta Martinovska Bande UPRAVLJANE ROBOTA PREKO INTERNETA	173
Isak Karabegović, Ermin Husak, Milena Đukanović APLIKACIJA INTELIGENTNIH SISTEMA-ROBOTA	177
Aleksandar Sokolovski, Sašo Gelev UPOTREBA GPU U SISTEMIMA ZA DETEKCIJU E-MAIL SPAM-A I IDS	181
Aleksandar Vučeraković PRIMJER UPRAVLJANJA RAČUNARSKIM SISTEMOM PUTEM GRUPNIH POLISA	185
Igor Miljanić PRIMJER ADMINISTRIRANJA HETEROGENOG RAČUNARSKOG SISTEMA RTCG	189
Zoran Veličković, Miloško Jevtović ADAPTACIJA IZGLEDA WEB STRANICE USLOVLJENA KLIJENSKIM SPECIFIČNOSTIMA U ASP .NET MVC 4 OKRUŽENJU	193
Stevan Šandi, Tomo Popović, Božo Krstajić ALATI ZA PODRŠKU MJERENJU SINHROFAZORA	197
Zoran Milivojević, Zoran Veličković, Dragiša Balanesković PROCENA INHARMONIČNOSTI KOPIJE ANTONIUS STRADIVARIUS VIOLINE	201
Biljana Chitkusheva Dimitrovska, Maja Kukusheva, Vlatko Chingoski LTspice IV KAO EDUKATIVNO SREDSTVO U NASTAVI ANALIZE ELEKTRICNIH KOLA	205
Petar Radunović, Tijana Vujičić, Ivan Knežević POREĐENJE FUNKCIONALNOG I IMPERATIVNOG PRISTUPA PROGRAMIRANJU	209
Jelena Ljucović, Ivana Ognjanović, Ramo Šendelj INTEGRACIJA ISTORIJSKIH PODATAKA U AHP ALGORITAM	213
Tijana Vujičić, Petar Radunović, Ivan Knežević KOMPARATIVNA ANALIZA NOSQL I SQL BAZA PODATAKA, NA PRIMJERU DATOMICA I MSSQL-A	217
Dragan Vidaković, Dusko Parezanović KRIPTOSISTEMI JAVNOG KLJUČA I GOLDBAHOVA PRETPOSTAVKA	221
Srđan Kadić, Milenko Mosurović POBOLJŠANA METODA VALIDACIJE 3N+1 HIPOTEZE PUTEM TRANSFORMACIJA	224
Stevan Šćepanović, Marko Grebović PLANIRANJE OBLASTI POKRIVENOSTI WLAN MREŽA	228

Milica Medenica, Sanja Zuković, Andjela Draganić, Irena Orović, Srdjan Stanković POREĐENJE ALGORITAMA ZA CS REKONSTRUKCIJU SLIKE	232
Marko Asanović, Radovan Stojanović, Igor Đurović METODA DETEKCIJE VATRE U REALNOM VREMENU NA BAZI OBRADU SLIKE	236
Nikola Besić, Gabriel Vasile, Budimir Lutovac, Srđan Stanković, Dragan Filipović ANALIZA PERFORMANSI FastICA ALGORITMA PRIMIJENJENOG NA 2D SIGNAL	240
Bojan Prlinčević, Zoran Milivojević, Darko Brodić EFIKASNOST MDB ALGORITMA KOD FILTRIRANJA SLIKA SA VODENIM ŽIGOM	244
Ratko Ivković, Mile Petrović, Petar Spalević, Dragiša Miljković, Boris Gara UTICAJ LINEARNOG OSVETLJENJA NA NIVO DETALJA I ENTROPIJU SLIKE	248
Alija Dervić, Nedeljko Lekić PREPOZNAVANJE DUŽICE OKA I UPOTREBA BIOCAM VISTA FA2.....	252
Luka Čadenović UTICAJ PUNJENJA VOZILA NA ELEKTRIČNI POGON NA KVALITET ELEKTRIČNE ENERGIJE U DISTRIBUTIVNOJ MREŽI	256

LTspice IV KAO EDUKATIVNO SREDSTVO U NASTAVI ANALIZE ELEKTRICNIH KOLA LTspice IV AS EDUCATIONAL TOOL FOR TEACHING ELECTRICAL CIRCUIT ANALYSIS

Biljana Chitkusheva Dimitrovska, Maja Kukusheva, Vlatko Chingoski, *Faculty of Electrical Engineering, University „Goce Delcev“ – Stip*

Sadržaj: U ovom radu predlaže se uvođenje osnovnih koncepata računarskog program za analizu LTspice IV, kao neprocenjiv alat u nastavi studenata predmeta analize električnih kola. Za studente, korišćenje ovog softverskog alata pruža mogućnost da rade samostalne analize ili da provere dobivene rezultate svojih analiza i time budu sigurni da su te analize napravljene na ispravan način. Kompjuterske simulacije u LTspice IV su vrlo značajne u procesu učenja omogućavajući studentu da eksperimentiše sa promenama i da vidi efekte koje te promene imaju na električna kola. Posmatrajući rezultate svoje kompjuterske simulacije, bročano i osobito grafički, student može da stekne intuitivno shvatanje ponašanja električnih kola bez zastrašivanja od upotrebe apstraktne matematičke analize.

Abstract: This paper proposes the introduction of the basic concepts of computer analysis program LTspice IV, as an invaluable tool in teaching students subjects of electrical circuits analysis. For students, the use of this software tool provides the ability to do independent analysis or verification of the obtained results within their analysis and thus be sure that these analyzes are made in the right way. Computer simulations using LTspice IV are very important in the learning process allowing students to experiment with changes to see the effects that these changes have on an electrical circuits. Observing the results of their computer simulations, numerically and graphically in particular, the students can gain an intuitive understanding of the behavior electric circuits without intimidation from the use of abstract mathematical analysis.

1. INTRODUCTION

Having understood the fundamental laws of circuit theory (**Ohm's law** and **Kirchhoff's laws**), we are now prepared to apply these laws to develop two techniques for circuit analysis: **nodal analysis**, which is based on a systematic application of Kirchhoff's Current Law (**KCL**), and **branch analysis** which is based on Kirchhoff's Voltage Law (**KVL**).

With these two techniques, one can analyze any linear circuit by obtaining a set of simultaneous equations that are simultaneously solved to obtain the required values of currents and/or voltages. One method of solving simultaneous equations involves Cramer's rule or method of replacing variables, which yields the unknown variables. Another method of solving simultaneous equations is to use **LTspice**, or any other circuit simulation software, simulate the electric circuit and solve it for its parameters.

LTspice IV is a free SPICE (**Simulation Program for Integrated Circuit Emphasis**) simulator with schematic capture from *Linear Technology Ltd.* Linear Technology (**LT**) is one of the industry leaders in production of various analog and digital integrated circuits [2]. LT provides a complete set of SPICE models mostly using LT components. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltages and current sources, a few types of dependent or independent sources, transmission lines, switches, and several devices: including diodes, BJTs, JFETs, MOSFETs. Circuits with large number (*almost infinite*) number of components can be simulated. One can think of LTspice as a nodal network solver that outputs all node voltages and branch currents, or simulate their changes

for prescribed time period. One node must be named "0" (*the ground node or the reference node*) and this node is used as a reference node for calculations of all other node voltages.

2. SHEMATIC ENTRY IN LTSPICE IV

Simulation of an electric circuit using LTspice IV can be done in two steps:

- drawing (*editing*) or entering circuit using schematic capture, and
- defining the desired type of simulation and execute the simulation or performing circuit analysis.

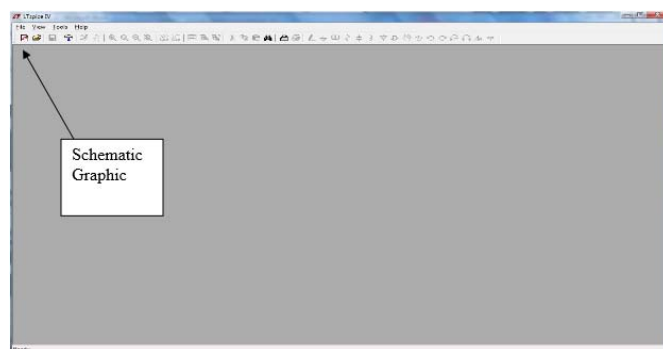


Fig. 1: Starting window in LTspice.

Drawing a circuit is easier when the grid mode is active. To activate the grid mode, click on **View** and select command **Show Grid**.

There are many buttons on the main menu for selection of basic circuit elements such as resistors, capacitors, inductors, diodes, wires, as well as a ground node (GND). The LTspice IV's **Main toolbar** is shown on Fig. 2.

3. CIRCUIT ANALYSIS WITH LTSPICE IV

As was already mentioned, the LTspice IV simulation software can be used as circuit analysis program which can easily determine all voltage and current values for any previously set circuit configuration and given numerical values for circuit components. We would demonstrate this, with several simple problems. These problems are usually solved by the students during their study of basic electrical circuit theory or any other basic or advance study subject that involves electric circuit analysis.

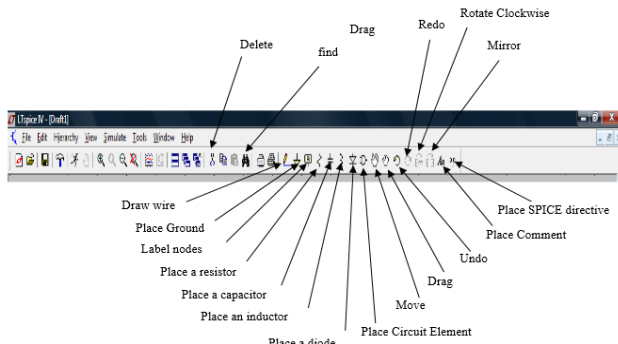


Fig. 2: Main toolbar of schematic editor for LTspice IV.

Example1: Find the branch currents I_1 , I_2 , and I_3 for a simple electric circuit show on Fig. 3.

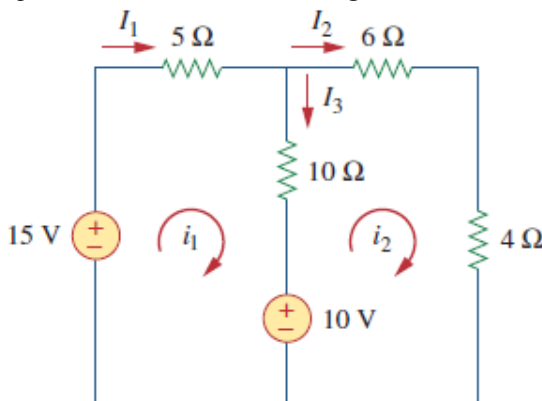


Fig. 3: Simple electric circuit as student task.

Usually students use some solving techniques such as contour currents or unknown node voltage analysis to find the unknown circuit parameters, voltages and/or currents.

Here, we could use the **Method of contour currents** to solve the above circuit for numerical values of all node voltages and branch currents. Consequently, one can obtain the following general system of equations:

$$\begin{aligned} R_{11} i_1 + R_{12} i_2 &= E_I \\ \pm R_{21} i_1 + R_{22} i_2 &= E_{II} \end{aligned}$$

Where, i_1 and i_2 are the unknown contour currents, R_{11} , R_{12} , R_{21} and R_{22} , are the equivalent contour resistances (*self and mutual contour resistance*), and E_I and E_{II} are the equivalent voltage source per contour. If we replace the numerical values given as in Fig.3, we could obtain the followings:

$$\begin{aligned} 15 i_1 - 10 i_2 &= 15 \\ -10 i_1 + 20 i_2 &= 10 \end{aligned}$$

Using the method of substitution to solve these system of two unknown currents with two equations, we get the values for the contour currents $i_1 = i_2 = 1A$. Thus, for the values of the unknown branch current we have:

$$I_1 = i_1 = 1A, \quad I_2 = i_2 = 1A, \quad I_3 = 0A$$

Next, we could show, how easy one can solve this electric circuit and get the numerical values and, if it is requested the graphic representation of all branch currents using the electric circuit solver and simulator, LTspice IV.

First, we open the LTspice program and start drawing (*editing*) the electrical circuit shown in Fig. 4. The analyzed electrical circuit contains two voltage generators and four resistors. After entering all circuit elements by clicking on the designated icons and setting their numerical values as parameters (*click on each component and set the desired parameters and values*), we place the circuit elements inside the program's workspace and finally connect the entire electric circuit with wires (*wiring*). The obtained electric circuit with its parameters shown inside the LTspice IV program's workspace is shown on Fig. 4.

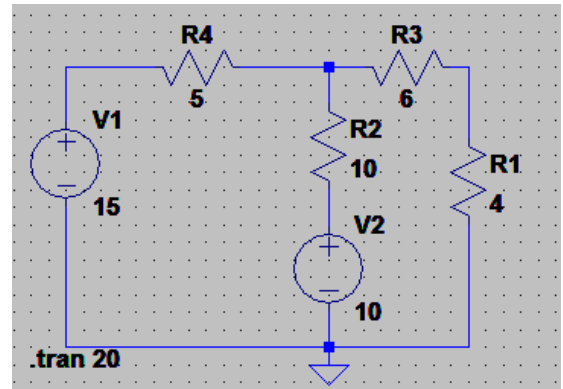


Fig. 4: Electric circuit built in LTspice IV.

At this point, our circuit is fully defined and we could continue to the next step, to solve or simulate the circuit behavior. The only relevant simulation for this electric circuit is a simple DC analysis, known in LTspice IV as **DC operating point**. This analysis we set by clicking on the icon **Simulate>Edit Simulation Cmd** from the top toolbar and select **DC op pnt**. This is called an **LTspice IV directive**.

To run a circuit (*execute the solver*), we can select from the file menu icon **Simulate** and **Run**, or we can simply click on the **Run** button from the main toolbar. Next, we could see a new window appearing, containing the simulation results, or in our case solution of the electric circuit in a simple table manner as shown in Fig. 5. As can be seen, the solution is exactly the same as in previous solution method using contour current method as a circuit solution method.

We must have in mind, that this was only a simple example, only two currents to solve for. However, if we have even a little more complex electric circuit, e.g. four or five contour currents, than the advantages of using LTspice IV for obtaining the unknown currents instead of solving by hand the same circuit by means of any other solution method are obviously large.

Next, we would present the advantages of utilizing LTspice IV software in the education of young students by

means of another example very often used in the classwork – the solution of an electric circuit using the **Thevenin's theorem**. Thevenin's theorem provides a solution technique by which one, usually called *the fixed part of circuit*, is replaced by an equivalent circuit [1]. Our major concern using this technique is to find the **Thevenin equivalent voltage** V_{TH} and the **Thevenin equivalent resistance** R_{TH} .

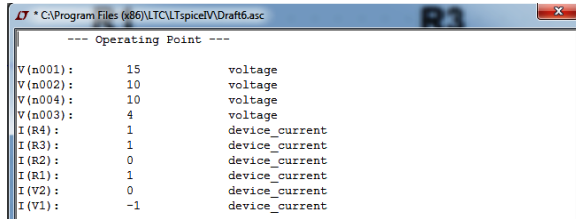


Fig. 5: Simulation (solution) results file.

Example 2: Find the equivalent Thevenin circuit of the circuit shown in Fig. 6 to the left of the terminals *a-b*. Then, find the current through the resistance $R_L = 6,16 \Omega$.

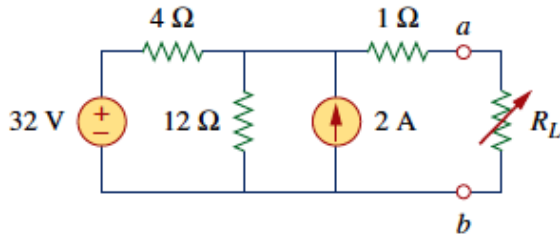


Fig. 6: Electric circuit for the Example 2.

We find equivalent resistance R_{TH} by turning off the 32V Voltage source (replacing it with the short circuit), and the Current source of 2A (replacing it with an open circuit).

Thus,

$$R_{TH} = 4 \parallel 12 + \frac{4 \cdot 12}{4 + 12} + 1 = 4\Omega$$

To find the equivalent voltage V_{TH} , we consider the electric circuit given on Fig. 7.

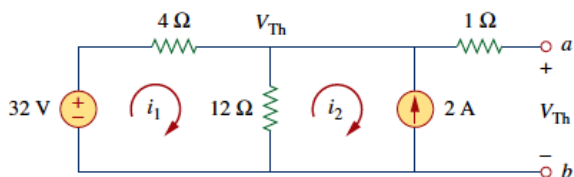


Fig. 7: Modified circuit to obtain the equivalent voltage V_{TH} .

Applying mesh analysis for the two loops, we obtain:

$$-32 + 4 i_1 + 12 (i_1 - i_2) = 0.$$

Solving for i_1 , we get $i_1 = 0,5 A$. Thus,

$$V_{TH} = 12 (0,5 + 2,0) = 30 V.$$

Respectively, the Thevenin equivalent circuit for the original electric circuit given on Fig. 6, get the configuration shown on Fig. 8.

The current through the R_L is:

$$I_L = \frac{V_{TH}}{R_{TH} + R_L} = \frac{30}{4 + R_L}$$

When $R_L = 6\Omega$, $I_L = \frac{30}{10} = 3A$. When $R_L = 16\Omega$, $I_L = 1,5A$.

If we built this electrical circuit in LTspice IV, the current through the R_L is determining in a very simple way without any calculations and complex methods.

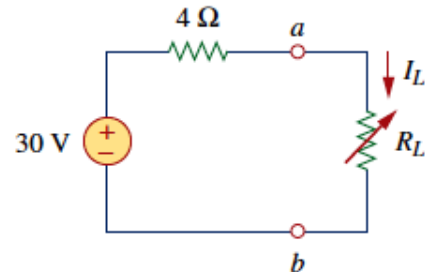


Fig. 8: Thevenin equivalent circuit for Example 2.

Using the same methodology described for the previous Example 1, one can draw the electric circuit of Example 2 in LTspice IV (see Fig. 9 – above).

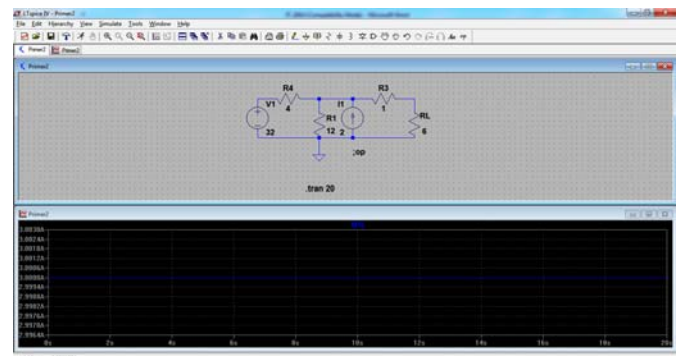


Fig. 9: Electric circuit of Example 2, built in LTspice IV (above), and the solution for the unknown current (below).

First, we could obtain the values of the current through resistor $R_L = 6\Omega$. For this value of R_L , we simulate the electric circuit shown in Fig. 6. After simulation, we get the results in output text file (.op file).

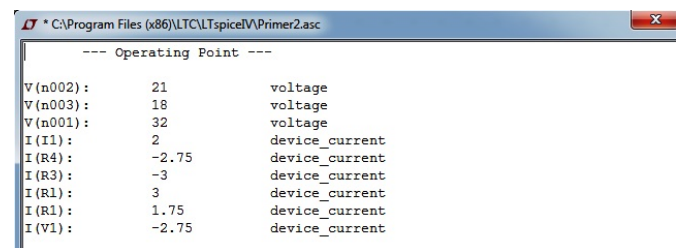


Fig. 10: Obtained results for $R_L = 6\Omega$.

LTspice has a user-friendly interface to show the results of simulations in two ways. One way is to display the results in a text file (.op file) (see Fig. 10), and the other way is using graphic display, as shown in Fig. 9 – below.

The third example of the advantages that simulation program LTspice IV provides for students and teachers in solving the subjects of electric circuit analysis, concerns about time-varying electric circuit analysis.

Example 3: Determine the time variation of the $V_0(t)$ in the circuit given on Fig. 11.

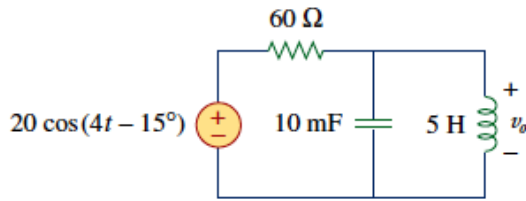


Fig. 11: Time-varying electric circuit.

To do the circuit analysis in frequency domain, we must first transform the time domain circuit, and its parameters into the phasor domain equivalent circuit. This transformation produces:

$$V_s(t) = 20 \cos(4t - 15^\circ) \Rightarrow V_s = 20 \angle -15^\circ, \omega = 4$$

$Z_1 = 60 \Omega$ is a impedance of the 60Ω resistor and $Z_2 = \frac{1}{j\omega C} \parallel j\omega L = j100 \Omega$ is a impedance of the parallel combination of the 10 mF capacitor and the 5 H inductor. Using the voltage-division principle, one can obtain:

$$V_0 = \frac{Z_2}{Z_1 + Z_2} V_s = 17.15 \angle 15.96^\circ \text{ V}$$

Now, we can convert this back to the time domain and obtain the final time-varying voltage value:

$$V_0(t) = 17.15 \cos(4t + 15.96^\circ) \text{ V.}$$

Let us now, solve the same problem using LTspice IV. First, we built the electrical circuit shown in Fig. 1. The electrical circuit contains one sinusoidal voltage source, resistor, capacitor and inductor, all of them with their respective numerical values. For the passive components (R, L, C), point at the component and do a right click. A dialogue box will appear and we could type the numerical value of the component. For the voltage supply, we do a right click on it and then click on the button **Advanced**. This enables us to set a variety of different voltage source waveforms. The one that we need for this analysis is on the left side of the dialogue box, as shown in Fig. 12.

Next, we do simulation of the circuit by clicking **Simulate > Edit Simulation Cmd** and type the **Stop time**. Finally, as a result we get the time changes of the unknown voltage $V_0(t)$ shown on Fig. 13. Later, various analysis can be performed of the obtained results such as reading of the frequency, max values, min values, attenuation factor, time constant, etc.

4. CONCLUSION

In this paper, we present how electric simulation program LTspice IV can be utilized to support teachers in education of students on the subject of electric circuit analysis.

We showed that LTspice is easy to use, very flexible and highly fast and accurate software for teaching electric circuit analysis. Three simple examples are given in order to provide

step-by-step insight how easy and resourceful could be this program for education of future electrical engineers. In addition to its solver capabilities, the program provides advanced graphic routines to visually grasp the nature of the process that happens inside simple, and especially very complex time and frequency varying electric circuits.

In such cases, use of LTspice simulation software allows for modeling of electric circuits and is an invaluable analysis tool. LTspice simulation software engages the user by integrating them into the learning experience. These kinds of interactions actively engage learners to analyze, synthesize, organize, and evaluate content and result in learners constructing their own knowledge.

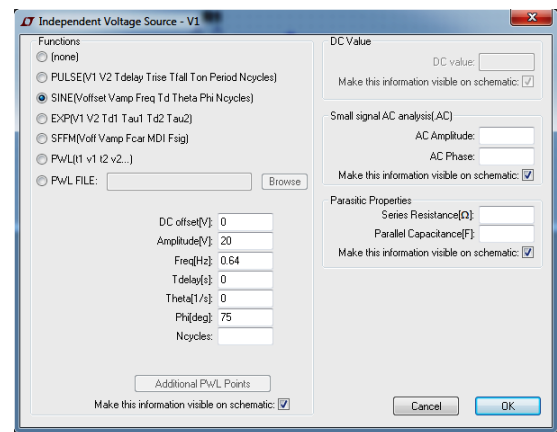


Fig. 12: Input screen for sinusoidal voltage source.

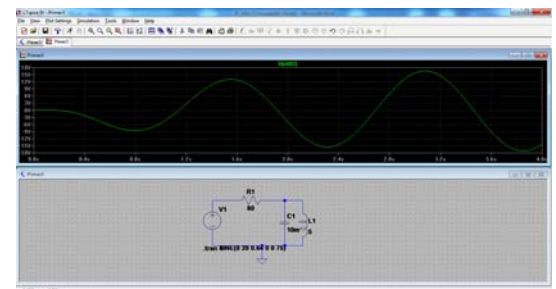


Fig. 13: Obtained time-varying function of voltage $V_0(t)$.

REFERENCES

- [1] Charles K. Alexander, Matthew N. O. Sadiku, "Fundamentals of Electrical Circuits," 5th ed., The McGraw - Hill Companies, New York, 2013.
- [2] LTspice IV tutorial, www.linear.com/ltspice
- [3] Влатко Чингоски, „Основи на CAD/CAM во електротехника,“ University „Goce Delcev“, Stip, 2013.
- [4] Susan A. Riedel and James W. Nilsson, "Introduction to Pspice for electric circuits," Pearson Education, Inc. 2007.

CIP - Каталогизација у публикацији
Национална библиотека Црне Горе, Цетиње

ISBN 978-86-85775-15-4
COBISS.CG-ID 24749840

ISBN 978-86-85775-15-4



9 788685 775154 >